EZ-ROUTE

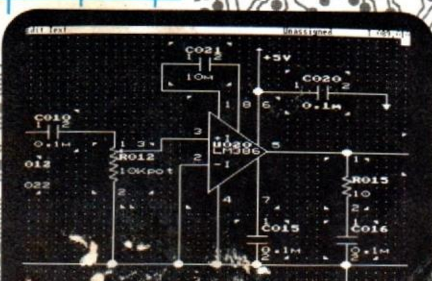
Integrated CAE Software Tools For Professionals

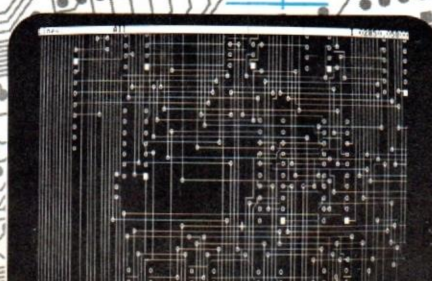
SPECIFIC INSTRUMENTS LTD
5115 EAST GEORGIA STREET
BURNABY, B.C. V5B 1V2

BILL JENKINS 604-291-6171

EZ-ROUTE

Schematic Capture
PC Board Layout
Automatic Router





EZ-ROUTE Evaluation Package

Software for Electronic Schematic
and
Printed Circuit Board Layout

Notice

The EZ-ROUTE system was developed by Phil Bolin, who wishes to thank Gail Bolin, Raj Shah, and Maggie Saad for their assistance.

Specifications are subject to change without notice.

When you have finished with the evaluation kit, please pass it on to a friend. The software in the evaluation kit is not copy protected.

To order call **1-800-9PCFREE.**

October 1988, 1st printing September 1989, 2nd printing

Advanced Microcomputer Systems, Inc.
1321 N.W. 65 Place
Fort Lauderdale, FL 33309 USA

EZ-ROUTE Evaluation Package
Table of contents

List of Figures

1 EZ-ROUTE Evaluation Package

1.1 Introduction

The purpose of this package is to help you evaluate the schematic capture, PC board layout, and auto-routing software of the EZ-ROUTE system. This evaluation package is divided into several sections. The sections are arranged in order of increasing detail. The first section gives a general overview of the entire EZ-ROUTE system. Other sections include specifications, software installation, self-running demonstrations, and hands-on tutorials. The final section gives information on how to order EZ-ROUTE products.

1.2 Contents of the Evaluation Package

The evaluation package contains special versions of the EZ-ROUTE schematic editor, PC board layout editor, and PC board auto-router. Also included are the schematic and PC board hard copy utilities. The package is designed to give you hands-on experience using the software. We think this package will convince you that this system can create and maintain PC designs easily and efficiently. This is a cost effective system.

If, after purchasing the EZ-ROUTE product, you are not satisfied with its performance, you may return it within 30 days for a full and prompt refund of the purchase price.

1.3 Limitations of the Evaluation Package

The programs furnished in the evaluation package function exactly like the real version except that you will not be able to save your designs to a disk file. The library of parts in the evaluation package contains only the parts and symbols required in the demos and tutorials. The real version comes with complete libraries of over 1000 symbol/part combinations. The evaluation package does not contain all the utilities of the complete EZ-ROUTE system. The next section describes the entire EZ-ROUTE system as it is delivered.

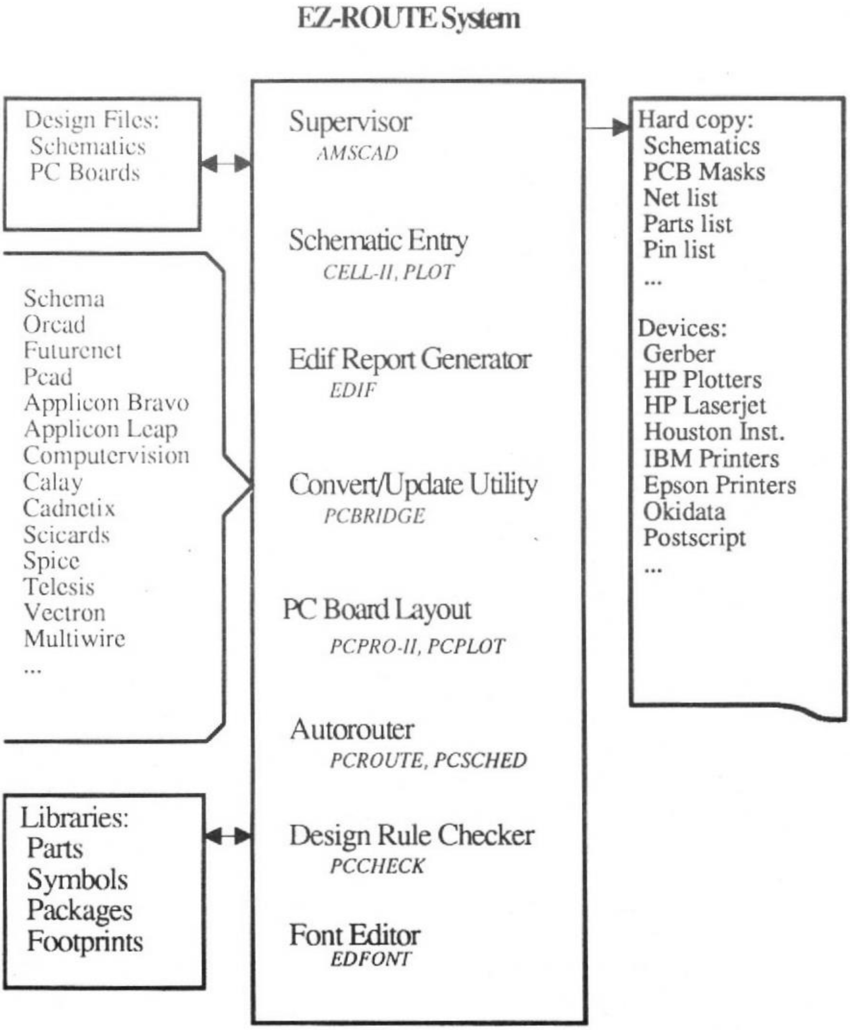


Figure 1-1. EZ-ROUTE Integrated System

2 Overview of the EZ-ROUTE System

This section of the evaluation package is designed to give you an overview of the entire EZ-ROUTE system. A description of each of the major components, and how they interact with each other is presented.

2.1 EZ-ROUTE Supervisor Program

The EZ-ROUTE supervisor program, AMSCAD, provides a smooth flow between the major components of the layout system. With this program, you do not need to remember or type program names. You select the schematic editor, PC board editor, or run demonstrations and other programs directly from menus. After running a program, you will return to the AMSCAD supervisor, ready to perform the next step in your layout.

Although the AMSCAD program provides an easy way to use the layout system, it is not required. All the programs in the layout system may be run directly without the use of the supervisor program. Also, the major subsystems of the EZ-ROUTE system are sold separately.

The supervisor program, as well as all the programs in the EZ-ROUTE family, have a consistent and easy to use interface. Selections are made from pull-down and pop-up menus. On line, context sensitive help is always available by pressing the F1 function key. Selection of file names is presented in menu form. The interface is object oriented. That is, you do not have to "back out" of the current operation to select another command or make option settings. You always just indicate what you want to do next. We know of <u>no other competitor</u> that has this modern, state of the art, type of user interface.

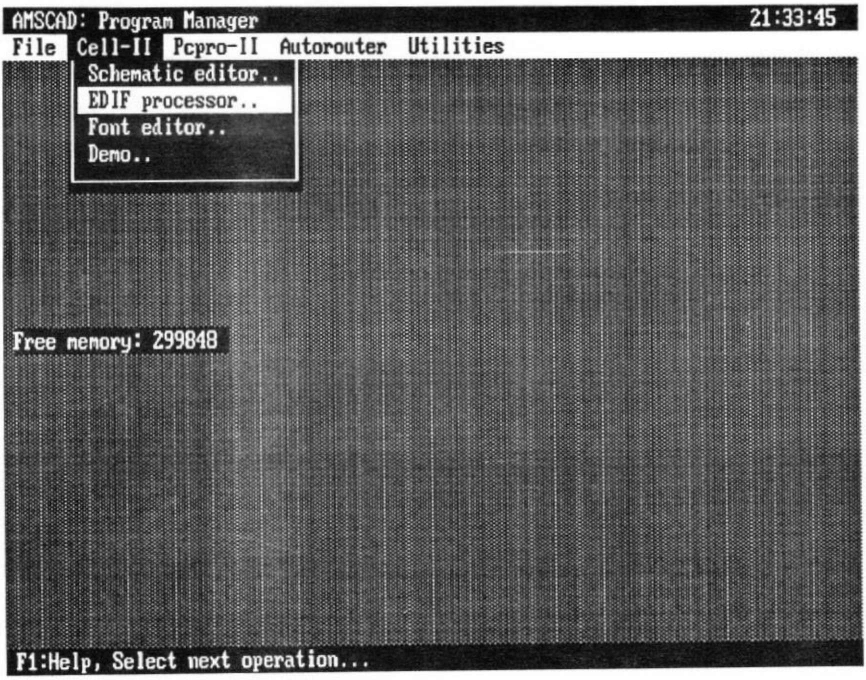


Figure 2-1. EZ-ROUTE Supervisor

A brief description of the functions provided by the supervisor program is given in the following sections. General descriptions are given here. Complete program specifications and limitations are given at the end of this section.

2.2 File Menu

The functions on the file menu allow you to perform general housekeeping operations, such as changing the working directory, entering a DOS shell, and exiting the program.

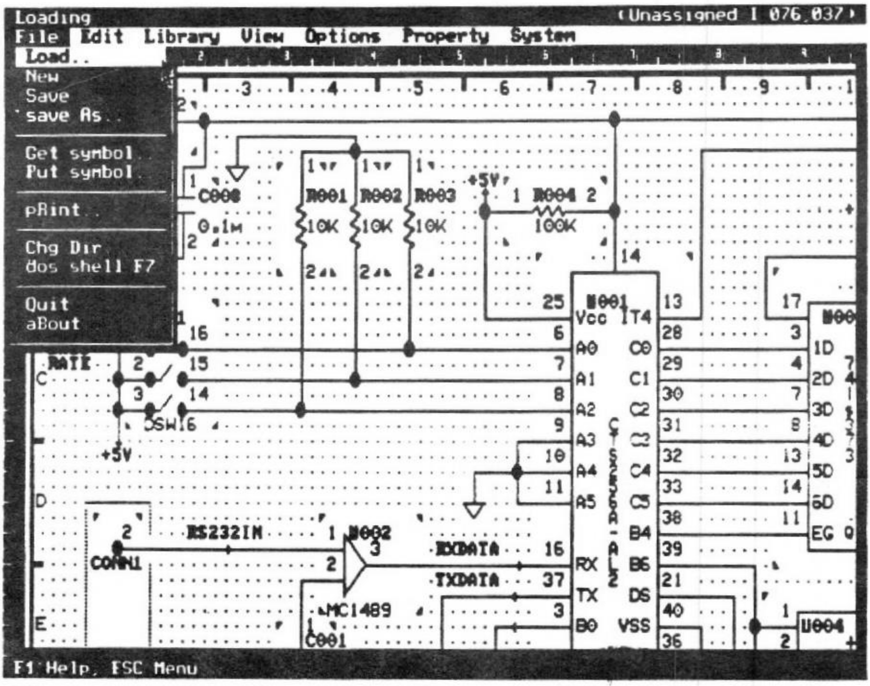


Figure 2-2. CELL-II Schematic Editor

2.3 Schematic Entry

The CELL-II menu provides access to the programs in the schematic layout system.

The **Schematic Editor** is used to create, edit, and maintain schematic drawings. The editor can also be used to draw part symbols, and to maintain the part, package, and symbol libraries.

The **EDIF Processor** generates net, pin, part, and numerous other reports from schematic drawings, including the initial placement of a PC board for use by the PC layout system.

The **Font Editor** allows you to customize or design your own character fonts for use on schematic drawings.

The **Demo** function is used to start a self-running demonstration of the schematic layout system.

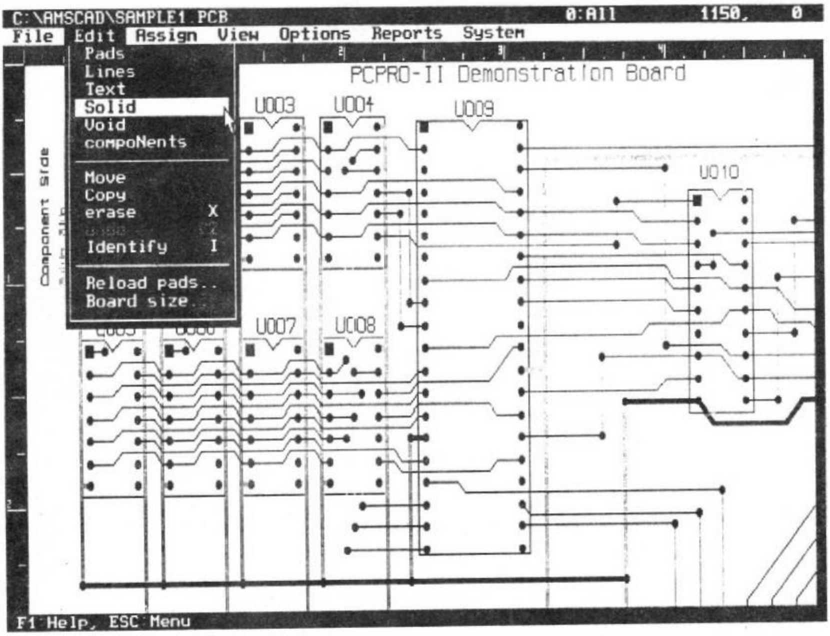


Figure 2-3. PCPRO-II PC Board Editor

2.4 PC Board Layout System

The PCPRO-II menu provides access to the programs in the PC board layout system.

The **PC Board Editor** is used to create, edit, and maintain the layout masks for PC board designs. The editor is also used to make PC board footprint symbols, and to print net lists, part lists, and other PC board specific reports.

The **Print/plot** program generates hard copy output of PC board masks. Popular printers, plotters, and photo- plotters are supported.

The **Font Editor** allows you to customize or design your own characters for use on a board layout, such as the text used on the silk-screen mask.

The **Demo** function starts a self-running demonstration of the PC board layout system.

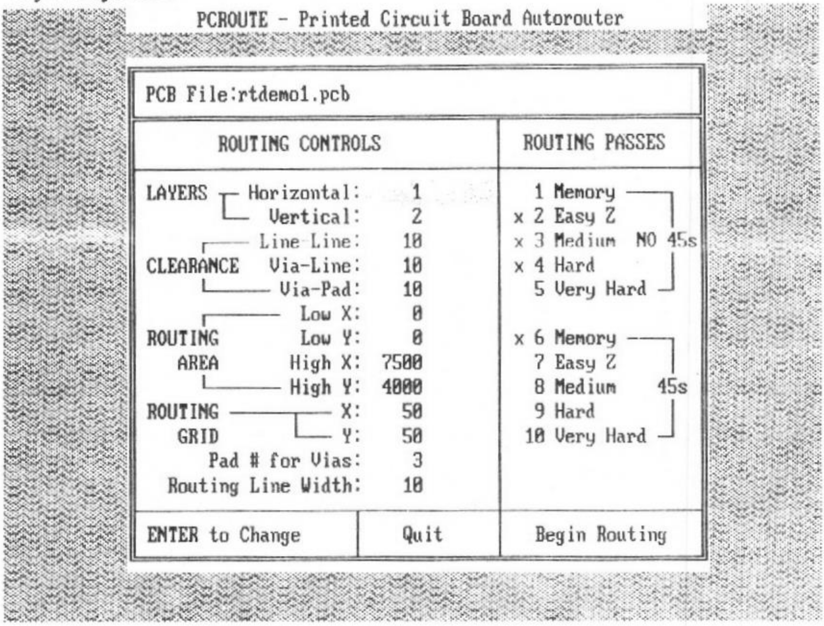


Figure 2-4. EZ-ROUTE Auto-router

2.5 Auto-router

The Auto-router menu provides access to the functions associated with the automatic routing of the electrical connections of a PC board.

The **Schedule Routes** function is used to automatically generate the routing schedule for a PC board. The schedule is normally used by the auto-router, but may also be used to assist you in manual layout by showing which points on a PC board must be electrically connected.

The **Auto-router** uses artificial intelligence to automatically route the electrical connections of a PC board. The auto-router will preserve any nets that you wish to pre-route manually. Also, the auto-router provides for being interrupted, editing the board with the editor, and restarting the router.

The **Demo** function starts a self-running demonstration of the auto-router program.

2.6 Utilities

The utility functions are used to convert data from other popular CAD systems to the EZ-ROUTE system (and vice/versa), and to verify that a PC board layout agrees with the schematic drawings and design rules.

The **PC Bridge** program provides a "bridge" between the EZ-ROUTE system and other popular CAD systems. This program is also used to verify and/or update a PC board's design data when changes are made to the schematics for the board.

The **PC Check** program checks a PC board layout for agreement with physical design rules. Such things as clearances of lines to pads, lines to lines, lines to vias, as well as numerous other checks are performed.

The **Smartwork Convert** function converts a Smartwork PC board file to an EZ-ROUTE board file.

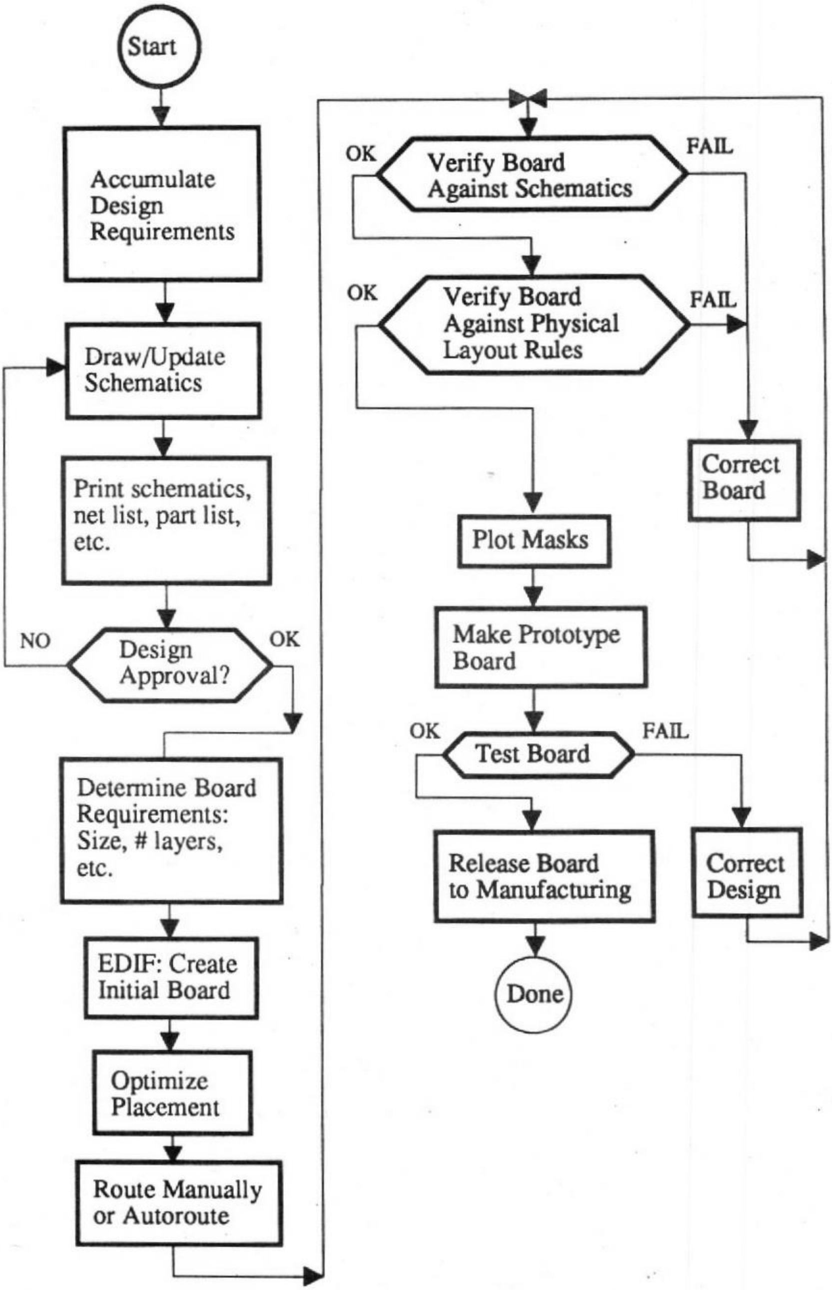


Figure 2-5. EZ-ROUTE Work Flow

3 Specifications

The following is a summary of the specifications for the EZ-ROUTE system.

3.1 System Wide Features

- Runs on IBM PC, IBM-XT, IBM-AT, IBM-PS/2 and compatible computers.
- May be used with or without a mouse device.
- State-of-the-art user interface with pull-down menus and dialog boxes.
- On-line and context sensitive help.
- Complete user manuals with tutorials for all the major subsystems.
- Video support for CGA, EGA, VGA, and Hercules graphic cards.
- Hard copy support for Epson, Hewlett-Packard, IBM, and Gerber printers/plotters. For an exact list, refer to the enclosed specification list.

3.2 Schematic Entry

- Sheet sizes A through E, and custom.
- Multiple sheets per design.
- Simple move, copy, erase, and draw operations.
- Drag symbols while keeping net line connections.
- Modern symbol/part libraries with symbol, part, and packaging information.
- Multiple symbols may be assigned to same part.
- Users may design their own symbols, and create their own symbol, part, and package libraries.
- EDIF processor generates EDIF, net, pin, part, PCPRO, Futurenet, and Redac reports and tests the schematics for errors. No need to put power and ground pins on the schematics, they can automatically be generated from the part library data.

3.3 PC Board Layout

- Boards up to 32 inches square, 255 masks.
- User selectable cursor layout grid. Items are not limited to fixed grid. Up to one mil resolution.
- Lines 0.001 to 0.255 inches wide.
- Up to 25 different pad shapes and sizes per board. Users may define their own pad shapes. Different layers of same pad may have different shapes.
- Up to four text fonts, font editor included.
- Detailed reports for net list, parts list, connectivity check, board summary of pads used, widths used, etc.
- Support for power and ground planes, with or without thermal relief.
- Hard copy output allows queuing of up to ten plots.
- Built-in plot panelization for plotting multiple layers per plot.
- Built-in net list name editor.
- Built-in "rat's net" prompting to aid in optimal component placement and manual routing of nets.

3.4 PC Board Auto-router

- Inputs and outputs a board file directly compatible withe the PC board editor.
- Routes multi-layer boards, two layers at a time.
- Routes boards with different lines width, such as wide lines for power and ground, and normal lines for the other nets.
- Router respects nets pre-routed by the user.
- Router may be interrupted, board edited with PC board editor, and restarted to finish the routing.
- User selectable routing parameters for line width and clearances.
- User selectable routing strategies.
- Visual display of the routing progress.
- Routing schedule may be generated automatically, by hand, or any combination of the two.
- Routing schedule may specify routing to a specific layer. This is required to route to edge connectors and SMD devices correctly.
- Uses a proprietary, state-of-the-art, artificial intelligence algorithm to route the nets. Routing completion rates are matched only by mainframe routers costing much more and requiring very expensive

equipment.

3.5 PC Bridge

- Can create/update an EZ-ROUTE PC board from the following net/pin list formats: EZ-ROUTE, Schema, Orcad, Futurenet, Pcad.
- Can create a net/pin list from an EZ-ROUTE PC board in the following formats: EZ-ROUTE, Schema, Orcad, Futurenet, Pcad, Applicon Bravo, Applicon Leap, Computervison, Calay, Cadnetix, Scicards, Spice, Telesis, Vectron, and Multiwire.

3.6 PC Check: Design Rule Checker

- Checks for net connectivity. Are all points which should be connected actually connected to each other?
- Checks for nets improperly connected (shorts).
- Checks for pad-to-pad, pad-to-via, pad-to-line, via-to-line, and line-to-line clearances.
- User selectable clearance values.
- Component, solder side, and inner layers may have different clearance rules.

3.7 User Support

- <u>Free</u> telephone support during normal business hours. You pay only for the call.
- If you sign up for our yearly update service, you will receive all scheduled program revisions as they become available.
- If you do not have the yearly update service, you can get the latest revision of any EZ-ROUTE program for a small update fee.

4 Hardware Requirements

The following hardware is required to use the EZ-ROUTE system.

Computer: An IBM or compatible computer with a minimum of 512KB. A hard disk is highly recommended, but floppies may be used.

Graphics: An EGA, CGA, or Hercules graphics card is required.

Mouse: Microsoft, Logitech, Torrington, Mouse Systems mouse, or compatible mouse is recommended, but not required.

Printer: A printer/plotter is not required by the evaluation package. For the real system, a printer is recommended for hard copy check plots and reports. Access to a pen or photo-plotter is recommended for final output of PC board masks.

5 Installation of Software

The EZ-ROUTE software may be used directly from the diskettes, or installed on a hard disk. You will get a much better feel for how the software works as an integrated design system by installing it on a hard drive. The following procedure may be used to install the software on the *C* hard drive from the *A* floppy drive. Substitute appropriate drive letters into the procedure if your system is different.

First, place the disk labeled **AMSCAD\PCROUTE** into the A drive.

```
> A:            (switch to A drive)
> CD \          (get to root directory)
> INSTALL C     (install on C drive)
```

This will begin the installation of the software on the hard drive. When prompted, remove the current floppy disk and replace it with the one indicated until the installation is complete.

The EZ-ROUTE software is designed to automatically detect the type of video graphic card in your system. On some clone systems, it may not be able to detect the video adaptor type correctly. If you have any problems running the software, or if you wish to explicitly set the video type, perform one of the following steps which corresponds to your system:

`> AMSCAD /AUTO`	*(Try automatic)*
`> AMSCAD /EGA`	*(EGA adaptor)*
`> AMSCAD /CGA`	*(CGA adaptor)*
`> AMSCAD /HGO`	*(Hercules adaptor, HG1-HG3 also available)*

This will get you up and running. When you exit the program, with the File/Quit command, you will be asked if you want to save the current configuration. Answer Yes. You may now run any of the programs by just entering the program name. The video type does not need to be specified.

The EZ-ROUTE programs will automatically detect if you have a mouse installed. In order to use the mouse, the mouse driver must be loaded prior to running the EZ-ROUTE programs, usually from the AUTOEXEC.BAT file. To install a mouse driver, refer to the installation instructions that came with your mouse.

6 Self-Running Demos

The evaluation package has self-running demos for the schematic editor, PC board editor, and the auto-router. When a demo is running, it is "playing back" the user input of an actual session. Therefore, the demos reflect exactly how the programs really work, and are not a contrived slide show as done by some other companies.

6.1 Running from an Installed System

If you have installed the software on your hard drive, enter:

`> C:`	*(switch to C drive*
`> CD \AMSCAD`	*if not already there)*
`> AMSCAD`	*(run program)*

to run the AMSCAD supervisor program. The program will display an action-bar with pull-down menus. Use the right and left arrow keys, or the mouse, to pull-down other menus. To run any of the demos listed on the menus, move the highlight bar with the up and down arrow keys to the desired demo. Press the *ENTER* key (or the *left mouse button*) to begin the demo. Be sure to run a demo from each of the menus (schematic, PC layout, and auto-router).

6.2 Running from Floppy Disks

If you wish to run the self-running demonstrations directly from the floppy disks, place the desired evaluation disk in the floppy drive and follow the printed instructions on the reference card included with that particular disk.

6.3 Controlling a Demo

While a demo is running, you may pause the demo by holding down the *ALT* key. The demo will continue when you let up on the *ALT* key.

You may stop a demo before it reaches the end by holding down <u>both</u> shift keys at the same time.

7 Tutorials

7.1 Introduction

This section presents two tutorials to give you actual hands-on experience. The first tutorial leads you through the creation of a simple schematic. The second tutorial is used to lay out a PC board of the circuit designed in the first tutorial.

7.2 Manual Notation

The following notation is used throughout the tutorials.

The notation "XXX" indicates to type the characters shown between the quotes. For example, "XYZ" indicates to type the characters X, Y, and Z. Do <u>not</u> type the quotes.

The notation, *XXX*, indicates to press this single key. For example, *ENTER* means press the enter key while *G* indicates to press the G key. Unless otherwise noted, either upper or lower case may be used.

The notation, <u>XXX</u>, indicates an item on a menu, or is used to emphasize particular words.

The notation, (xxx,yyy), indicates a coordinate. For example, (10,15) indicates the horizontal position is 10 and the vertical position is 15.

The tutorial steps are listed in terms of the keyboard. If you are configured with a mouse, feel free to use it. The left button of the mouse is equivalent to the *ENTER* key, while the right button is the *ESC* key. The middle button, if present, may be used as the *MARK* key (F10). Any combination of the keyboard and mouse may be used.

Initially, great detail is given. As we proceed through the tutorial you will learn the basics, less detail will be given.

The print/plot utilities are included in the evaluation kit. If you will be using a plotter connected to a serial port, refer to appendix A for additional important information.

7.3 Schematic Tutorial

This tutorial leads you, step by step, through the creation of a simple schematic. If you have not yet installed the evaluation software, and you wish to do so, refer to the previous section for instructions.

7.4 Start Schematic Editor

The schematic editor may be selected from the AMSCAD menu, or run directly from the demo disk. To run directly from the demo disk, put the disk in drive A and enter:

`> A:`	*(make A drive the default)*
`> CD \`	*(switch to root directory)*
`> CELL`	*(run the schematic editor)*

If you have installed the software on a hard disk:

`> C:`	*(make C drive the default)*
`> CD \AMSCAD`	*(where cell is installed)*
`> AMSCAD`	*(run the EZ-ROUTE supervisor)*

Select <u>Schematic editor</u> from the CELL-II menu.

7.5 Loading a Schematic

To get a feel for the system, we will first load a sample schematic file. We will then learn how to move about the schematic on the display.

Use the *up* and *down* arrow keys on the numeric key pad to move the menu pointer up and down. The *NumLck* key must be off.

Move the select indicator to the <u>Load</u> entry, and *ENTER*. This is referred to as selecting a menu entry.

We are now prompted for the name of a schematic file. At this point, we could enter the file name of a schematic if we knew it, but since we do not, press *ENTER*. Now, we are prompted for the name of a directory to be searched for schematic files. Press *ENTER* to use the local directory.

A menu of the schematic files on the current directory is now displayed. Select the VOICE1.DWG entry from the menu. This demonstration schematic will be loaded and displayed on the screen, along with the file menu.

7.6 Getting Help

Press the *F1* key to pop-up a window of help information. Notice that the help information is specific to the current menu item. Press *ESC* to remove the help window. Now, move the menu bar up and down to different items, selecting help on each one with the *F1* key. If more help information is available than can fit in the window, a slide-bar will appear on the right side of the window. Use the *PgUp* and *PgDn* keys to page through the help information.

Help is always available with the *F1* key.

7.7 Moving Around a Schematic

We will now learn how to move about a schematic on the display screen.

Press the *right arrow* key to pull-down the menu to the right. Repeat until you are on the View menu.

Move the select bar to out and select it. The display will be redrawn at a smaller scale. You will notice that the out menu item has *^PgDn* listed in the right margin. This indicates that this key may be used from the edit window to perform this same command. Let's try it. Now, hold the *Ctrl* key down and press the *PgDn* key to zoom in. Most menu items which have a key noted in the right margin may be executed directly from the edit window. Once you are more familiar with the program, using these "hot-keys" directly will improve your productivity, especially if you do not have a mouse.

Zoom back to the original scale by pressing the *Ctrl-PgUp* key twice.

The current coordinate of the cursor is displayed in the upper-right corner of the screen. Use the cursor keys to move the cursor to coordinate (10,10). Follow the track of the cursor on the screen, and watch the value in the upper-right corner change.

Holding the *Shift* key down and using the cursor keys will move the cursor 8 units at a time. Hold the *Shift* key down and use the *left* and *up* arrows to move quickly to coordinate (0,0), the upper-left corner of the board.

The display may be panned from the keyboard by bumping into one of the edit window edges. Let's pan right. Move the cursor with the *shift right arrow* key until it hits the right edge of the window. The portion of the schematic that was out of view to the right of the screen is now displayed.

With a mouse, you may also pan by placing the cursor in one of the windows that surround the edit window, such as one of the rulers. The cursor will change to an arrow. Press *ENTER* to pan in that direction.

Display the upper-left corner of the sheet by pressing *Ctrl-Home*.

7.8 Cursor Grid

The little dots that you see on the screen comprise the cursor grid. It is used to give the user reference points to assist in the schematic layout. It is not part of the schematic, and is not printed on hard copy output. The grid may be turned on or off. Type *V* to pull-down the view menu and type *G* to select grid. Notice that the grid has been removed from the display.

This completes a summary of how to start the CELL-II editor, load an existing schematic, view the schematic, and control the cursor grid. Type *ALT-G* to turn the grid back on.

7.9 Creating a New Schematic

Our assignment is to create a small board entirely from scratch. We must layout the schematic drawing for the circuit. This circuit is intentionally small; however, it demonstrates the basic steps by which any circuit, regardless of size, is created. A rough, hand-drawn sketch of the circuit is given in Figure 7-1. This is the way actual schematics normally start out. An experienced designer may use CELL-II to directly enter designs without the use of rough sketches.

Specific requirements supplied by the designer are as follows:

1. Use TTL logic.
2. PC Board size is 4.00" x 4.00".
3. Board has 2 signal layers, route with 12 mil lines.
4. Power and ground provided by power planes.

Detector Circuit

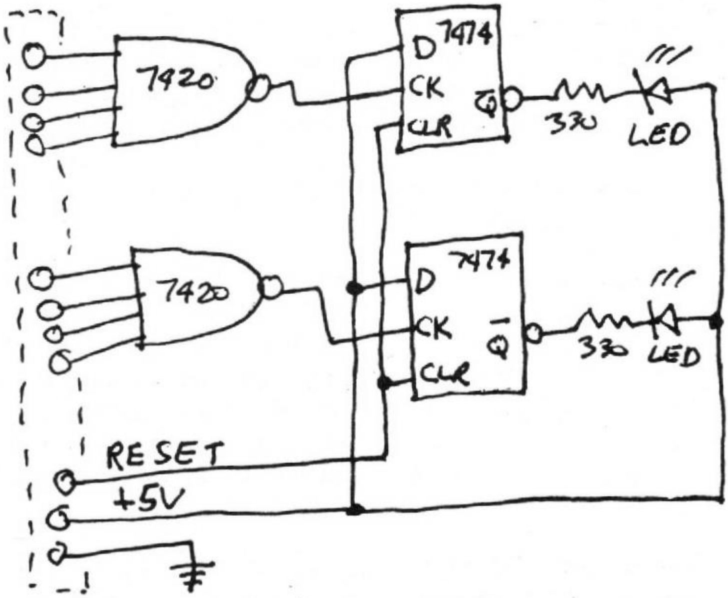


Figure 7-1. Rough Sketch of Circuit

We have all the information we need. We are now ready to begin our schematic. Go to the <u>File menu</u> (*F*) and select <u>New</u>. If you have made any changes to the schematic we have been viewing, you will be prompted whether or not you really want to abandon the changes. If prompted, type *Y* for yes.

We are presented with a menu of drawing sizes to choose from. Since we have a simple circuit, select the A size drawing. Now, we have a blank A size sheet to work on.

7.10 Placing the Parts

We are now ready to place the parts of our circuit on the sheet. Pull-down the *Library* menu and select *Load Part*. We will first place the four-input NAND gates on the sheet. Enter <u>7420</u> for the part name and press the <u>ENTER</u> key. The program will search the parts libraries for the indicated part. The cursor is now a rectangular area, indicating the size of the symbol for this part. Move the cursor to coordinate (8,5). <u>ENTER</u> to place the symbol at the point. Since this particular part has more that one gate, we are now prompted for which section (gate) of the part we want to use. Select the *A* section. Now we are prompted for the instance number to assign to the symbol. Just *ENTER* to accept the <u>U1</u> value. Now the symbol, with the pin numbers per the section and instance number we assigned are displayed.

Now, let's add the other gate. Move the cursor to (8,17) and *ENTER*. Accept the suggested section, <u>B</u>, and the instance number, <u>U1</u> with the *ENTER* key. Notice that the two NAND gates that we added to the schematic are really contained in the same physical part.

Now we will add the flip-flops. First, enter *ESC* to cancel adding any more of the 7420's. Now, select *Load Part* from the library menu. Enter <u>7474</u> for the part number. Move to (27,5) and *ENTER* to place the first flip-flop. Accept the suggested part section and instance number. Now, move to (27,17) and place the other half of the flip-flop. Isn't this much easier than drawing the parts every time?

Now we will add the 330 ohm resistors to the schematic. *ESC* from the previous part. Load the resistor part <u>R.330</u>. The period is part of the part name. Place this resistor value at coordinate (40,10) and (40,22) as <u>R3</u> and <u>R4</u>.

Now we will add the light-emitting diodes to the schematic. In this case, we will intentionally make a mistake, and recover from it. Load the part <u>LED-1</u>. Place the first LED at (52,8). Now look at the orientation of the LED on the screen. You will notice that the cathode end of the LED is on the wrong (right) side. This is not what we want. Press the

F10 key. Now, we have a list of possible symbols for this part. Select the *LED.HL* (HL for horizontal left). Now place the part again in the same coordinate (52,8). Since something is already here, we will be prompted if it is ok to place this symbol on top of what is already here. Answer yes for the override protection and overlay prompts. When prompted for the instance number, set it to D5 (the original D5 was wrong). Now, we have an LED pointed in the right direction. Now place the second LED, D6, at coordinate (52,20).

We now have all of our true parts placed. The pseudo-part, the connector, will be constructed "on the fly" later.

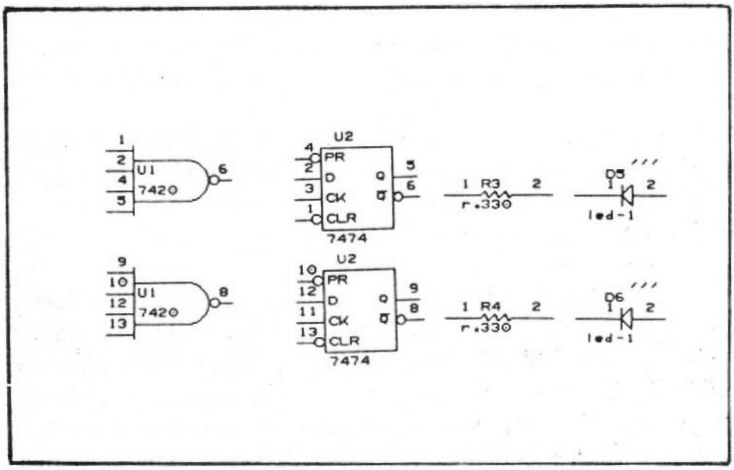


Figure 7-2. Parts Placed on Schematic

7.11 Wiring the Circuit

Since all of our parts are placed, we are now ready to wire them together. First, let's wire the connection from U1 pin 6 to U2 pin 11. From the Edit menu, select the Wire Net command. Move the cursor to (20,9). *ENTER* to begin the net line. Move to (21,9) and *ENTER* to draw a segment of the net. Move to (21,11) and *ENTER*. Move to (27,11), *ENTER* to draw the segment, and *ESC* to stop drawing this net's lines.

Now, let's make a similar connection from U1 pin 8 to U2 pin 11. Move to (20,21) and *ENTER* to start the net line. Move to (21,21), *ENTER*, (21,23), *ENTER*, (27,23), *ENTER*, and *ESC*.

Got the hang of it? In actual practice, the wiring process is even easier because you are drawing the lines by looking at the points that need connecting on the screen instead of working from a list of points as we are doing here. If you add a line in the wrong place, simply remove it by drawing another line over the wrong portion.

Now, connect the following points with a net line. Remember to use *ESC* to stop drawing one net before starting the next net line.

```
U2:6, R3:1   (38,11) to (40,11).
D5:1, R3:2   (49,11) to (52,11).
U2:8, R4:1   (38,23) to (40,23).
D6:1, R4:2   (49,23) to (52,23).
D5:2, D6:2   (60,11) to (63,11) to
             (63,23) to (61,23).
```

Now, move the cursor to (63,23). At this point, we want to add a connect dot and continue the net. *ENTER* to start the line. Now, without moving the cursor, *ENTER* again. This will add a connect dot. From this point, move to (63,30) and *ENTER*. Continue this net line to (23,30), (23,21), and into U2 pin 12 at (27,21). Now *ESC* and move back to (23,21). Add a connect dot here, and continue the net to (23,9) and end at U2 pin 2 (27,9). Move to (23,30) and add a connect dot. Continue the net to (0,30). This is actually two grids to the left of where we want to be; but, we are intentionally making this mistake so we can correct it later with the Drag command.

7.12 Making a Connector

At this point, let's assume that we do not have a part or symbol in the library for the connector. We will build the connector directly on our schematic. The procedure demonstrated here shows how to make a new symbol for a part. If we wanted to, we could store it into one of the libraries for use on other circuits. All symbols are made in this fashion - draw it once on a schematic and store it into a library.

At the current point (0,30), we will terminate the net with a connector (or off page reference) indicator and build a part for it. This is just like adding a connect dot, except *ENTER* one more time to convert the solid dot to an open dot. If we wanted to, we could *ENTER* one more time to remove the dot.

From the Edit menu, select the Copy command. Move to (0,30) and MARK (F10 or the middle mouse button) to begin marking an area. Now move to (7,30) and MARK again to complete marking the area. At this point, the area we want to copy is marked but has not yet been selected for copying. *ENTER* while the cursor in within the marked area to select it (where the cursor is now is just fine). The cursor now has the shape of the area we are about to copy. Move to (0,32) and *ENTER*. We have placed a copy of the connector lead we just drew at this point. Now, move to (0,28) and *ENTER* to place another connector lead here. Continue this procedure to copy the lead to the following coordinates:

```
(0,24)  U1:13
(0,22)  U1:12
(0,20)  U1:10
(0,18)  U1:9
(0,12)  U1:5
(0,10)  U1:4
(0,8)   U1:2
(0,6)   U1:1
```

ESC to stop copying. As we said before, the connector has been located two grids left of where we really want it. From the Edit menu, select the Drag command. Move to (0,5) and *MARK* to start marking an area. Move to (2,33) and *MARK* to complete the marked area. Select the area with *ENTER*. To get a better feel for how this command works, use the keyboard arrow keys for the next few steps, even if you have a mouse. Press the right arrow key. Notice how the entire marked area moved to the right. Press the right arrow key three more times. Now press the left arrow key two times to move it to the left. Notice that the net lines have been stretched and not broken. This is a very handy command. The area should now be two grids to the right of where it was before we dragged it. This is where we want it.

Now let's add the pin numbers for the connector. From the Edit menu select the Text command. Now type ALT-P, or use the mouse, to pull-down the Property menu. Notice that to pull-down a menu from the keyboard while in the text command, it is necessary to hold the ALT key down and type the menu select character. If you just typed the menu character, that character would be typed onto the schematic. *F9* may also be used to pull-down the property menu.

Select the Input property from this menu. Now move to (3,5) and type the character 1 for pin number one. Move to (3,7) and type 2 for pin number two. Proceed to label the connector points as follows:

```
3 at (3,9)
4 at (3,11)
5 at (3,17)
6 at (3,19)
7 at (3,21)
8 at (3,23)
9 at (3,27)
```

At this point, select the Power property from the property menu.

```
10 at (3,29)
11 at (3,31)
```

Now, we have all the connector pins identified with their pin numbers. It is important to note at this point that the property selection is important. We not only put the text characters on the schematic, but we also encoded the information that the characters represent input pins and power pins.

Now, we will add the connector instance (name) and part number. Since we are tight for space, we'll add the text vertically. Select the Instance property. From the Option menu (*ALT-O*) select the Vertical text option. Move to (1,12) and type "CONN". Now select the Part property. Move to (1,25) and type "EC.12". From the option menu, set the text mode back to Horizontal text.

Now, we will convert the connector we have just draw into a symbol. From the Edit menu, select the Symbolize command. Move to (1,4) and MARK. Move to (3,33) and MARK. ENTER to convert the area into a part symbol. Note that the unused area of the symbol has been converted to blanks, and the symbol corner designators have been added.

To make the connector stand out a little, we will add a little decoration around it. From the Edit menu select the Lines command, and from the Property menu, select the Unassigned property. Unassigned means this item has no circuit meaning, it is here just for artistic or notation purposes. From the Option menu, select the Dashed line type (shown graphically on the menu). Now, draw a line, just as we drew nets, between the following points.

```
(5,5) to (5,3) to (0,3) to (0,34) to
(5,34) to (5,33) stop.
(5,27) to (5,25) stop.
(5,17) to (5,13) stop.
```

Now, we will add the net names for the nets which have names. Select the Net Name command from the Edit menu. This works just like the text command we have already used, except the property is automatically set to Net for you. Add the following net names at the given locations:

```
I1 at (6,5).
I2 at (6,7).
I3 at (6,9).
I4 at (6,11).
I5 at (6,17).
I6 at (6,19).
I7 at (6,21).
I8 at (6,23).
RESET at (6,27).
+5V at (6,29).
```

Pin 11 of the connector is the ground net. We will show this by using the ground symbol. First, use the Edit/Wire Net command to extend the net line from connector pin 11 to (15,32) and down to (15,34). Now, select the Library/Get symbol command. Enter "GND" for the symbol name. Note that the ground symbol is a symbol and not a part. Place the symbol at (15,34). Answer yes to the overlay question. Now, this net is connected to ground. We could have accomplished the same effect by labeling the net as GND.

We need to complete the wiring for the RESET net. In drawing this net, we will demonstrate another way of drawing lines. Select the Edit/Wire net command and move to (27,13). Now, press *F2*. This enables the draw step mode. This method will work only from the keyboard. With the arrow keys, move left to (25,13), then down to (25,25), and right to (27,25). Notice how the line is drawn as the cursor moves. If you make a mistake, just move back over the line to undraw it. Press *F2* to switch back to move mode. Move to (25,25) and *ENTER* twice to add a connect dot. *F2* to enter step mode. Move down to (25,28) and left until the RESET net is completed. This method of drawing lines is very effective in tight spaces, and for correcting mistakes.

Now, we will complete our schematic by adding some notational text. Select the Edit/text command and the Unassigned property. Move to (23,32). MARK to establish a left margin. Enter the following text. Note, with the left margin present, just type the text like on a typewriter. ENTER functions like the carriage return.

Notes:
1. Layout on a 4" x 4" board.
2. Power and ground planes required.

Now, move to (25,2). Select Bold text from the Option menu. Type "Detector Circuit". Now, select the Edit/Lines command and the Double line type. Draw a line around the text we just added.

7.13 Saving the Schematic

This completes the tutorial schematic. Let's save it to disk. Select the File/Save as command. Enter MYDEMO for the name. ***The evaluation version of the program will not actually save the drawing.***

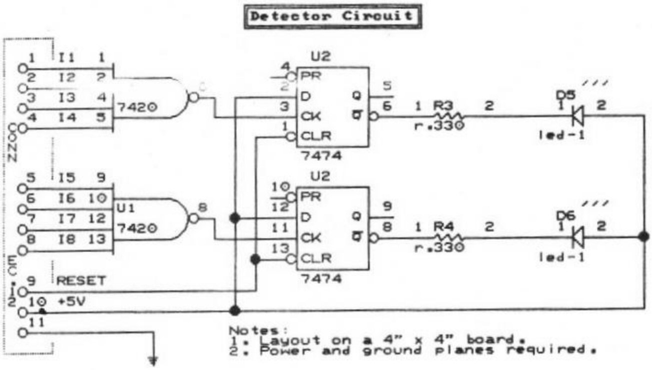


Figure 7-3. Finished Schematic

7.14 Exit the Editor

We are finished with the schematic for now. Select Quit on the File menu to exit the program.

7.15 Summary of Schematic Entry

Let us emphasize that the entry of the tutorial schematic may seem a little tedious. This is because we have stepped you through each of the steps at a very detailed level. In actual practice, you are choosing your own locations for parts, paths for the nets, and the order in which you wish to layout the circuit. Just follow the principles you have learned here and it will flow very smoothly.

8 PC Board Layout Tutorial

This tutorial leads you, step by step, through the creation of a simple PC board. The board corresponds to the schematic created in the previous tutorial. If you have not performed the schematic tutorial, please refer to sections 7.2 and 7.6. These sections explain the manual notation used in the tutorials.

Although we will be using the results from the schematic tutorial to lay out the tutorial board, it should be emphasized the PCPRO-II system may be used to lay out a board entirely from scratch, or from the net lists from several other CAD systems.

8.1 Start PC Board Editor

The PC board editor may be selected from the AMSCAD menu, or run directly from the demo disk. To run directly from the demo disk, put the disk in drive A and enter:

```
> A:          (make A drive the default)
> CD \        (change to root directory)
> PCPRO       (run PC editor program)
```

If you have installed the software on a hard disk:

```
> C:          (make C drive the default)
> CD \AMSCAD  (where installed)
> AMSCAD      (run the AMSCAD supervisor)
```

Select PC BOARD EDITOR from the PCPRO-II menu.

8.2 Loading a Board

To get a feel for the system, we will first load a sample board file on the release disk. We will then learn how to move about the board on the display.

Move the select indicator to the Load entry, and *ENTER*.

We are now prompted for the name of a board file. At this point we could enter the file name of a board if we knew it, but since we do not, press *ENTER*. Now, we are prompted for the name of a directory to be searched for board files. Press *ENTER* to use the local directory.

A menu of the board files on the current directory is now displayed. Select the SAMPLE1.PCB entry from the menu. This demonstration board will be loaded and displayed on the screen, along with the file menu.

8.3 Moving Around a Board

We will now learn how to move about a board on the display screen.

Press the *right arrow* key to pull-down the menu to the right. Repeat until you are on the View menu.

Move the select bar to in and select it. The display will be redrawn at a larger scale. Zoom in again. You will notice that the in menu item has ^PgUp listed in the right margin. This indicates that this key may be used from the edit window to perform this same command. Let's try it. Press the *ESC* key to roll-up the pull-down menu and switch to the edit window. Now, hold the *Ctrl* key down and press the *PgDn* key to zoom out. All menu items which have a key noted in the right margin may be executed directly from the edit window. Once you are more familiar with the program, using these "hot-keys" directly will improve your productivity, especially if you do not have a mouse.

It should be noted that if you pick the pan and zoom commands directly, you do not need to wait for the redraw to complete before entering another pan or zoom command. In this case, the current redraw will be stopped, and the next redraw will begin directly.

The current coordinate of the cursor is displayed in the upper-right corner of the screen. Use the cursor keys to move the cursor to coordinate (1900,900). Zoom in or out if necessary. Follow the track of the cursor on the screen, and watch the value in the upper-right corner change.

Function key *F8* is the locate-text command. This command is used to locate a text string on a board. Press *F8* and enter "U005" *ENTER* for the search (in the input window). The program will pan the display so that the located string is displayed. The position of the cursor is the coordinate of the text.

The display may also be panned from the keyboard by bumping into one of the edit window edges. Let's pan down. Move the cursor with the *down arrow* key until it hits the bottom edge of the window. The portion of the board that was below the bottom of the screen is now displayed.

With a mouse, you may also pan by placing the cursor in one of the windows that surround the edit window, such as one of the rulers. The cursor will change to an arrow. Press *ENTER* to pan in that direction.

Display the whole board by pressing *Ctrl-Home*.

Holding the *Shift* key down and using the cursor keys will move the cursor eight units at a time. Hold the *Shift* key down and use the *left* and *up* arrows to move quickly to coordinate (0,0), the upper-left corner of the board.

8.4 Selecting Layers for Display

We will now explore how to select the layers of information that are displayed on the screen. This is controlled by selecting Display layers from the view menu. To pull-down the view menu, press the *V* key. We could also have pressed *ESC* and then moved right or left to get to the view menu.

Select Display layers to pop-up the display layer control menu. You will notice that each layer of the board is listed along with its type and current display color. If the color is off (same color as the PC board background), then that layer will not be displayed. The actual colors and number of colors available will depend on your particular hardware. The display state of each layer is changed by pressing *ENTER* for a full menu of colors, or using the *right* and *left* arrow keys to toggle the display state. Move the menu pointer to the silk-screen layer and press the *ENTER* key. Now, select off. Proceed to turn all the layers except the Component and All layers to off. Exit from the menu by pressing the *ESC* key. The display will be redrawn per the new settings. Notice that only the component layer is displayed.

Now, let's display just the solder-side layer by entering *D*, set the component layer to off, set the solder-side layer to the color of your choice, and *ESC*. Only the solder-side is displayed.

8.5 Cursor Grid

The cursor grid is set to 50 by default. You have probably noticed that the cursor steps 50 units each time the arrow keys are pressed. Select the Grid entry from the View menu. Move the pointer to the X grid selection and *ENTER*. Enter "25" and *ENTER*. This sets the X cursor grid value to 25. Do the same for the Y grid. Now, press *ESC* to exit the grid menu. Move the cursor about the screen with the arrow keys. Notice that the cursor now moves in steps of 25.

This completes a summary of how to start the PCPRO-II editor, load an existing board, view the board, control the display of selected layers, and set the cursor grid.

8.6 Layout of a New Board

Our assignment is to lay out a small board which corresponds to the circuit we drew during the schematic tutorial. After drawing a schematic, the normal procedure is use the EDIF processor to print a parts list, a net list, and to create an initial PC board from the schematic data. Since the EDIF schematic processor is not included in the evaluation kit, the PC board file, MYPCB.PCB, is included. This file was produced by the EDIF processor, and is the initial PC board corresponding to the tutorial schematic circuit. The EDIF processor generates a board file which contains all the components and pin/net name assignments specified on the schematic.

This board is intentionally small; however, it demonstrates the steps by which any board, regardless of size, is created. The schematic, parts list, and net list produced by the EDIF processor are given in Figures 8-1 through 8-3.

Additional specifications are as follows:

1. Board size is 4.00" x 4.00".
2. Board has 2 signal layers,
   12 mil lines.
3. Power and ground provided by
   power planes.
4. Solder mask required.
5. Silk screen required.
6. Drill drawing required.

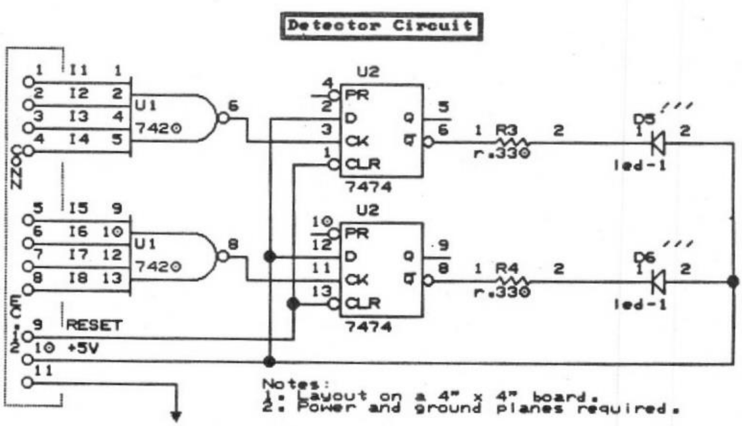


Figure 8-1. Schematic Drawing

```
Part List
---#--Part---Count--Instances-----
   1  ...        1  CONN
   2  320        2  R003  R004
   3  LED        2  D005  D006
   4  LS20       1  U001
   5  LS74       1  U002

Net List
---#-Net name---Nodes-------------
   1 #001       U002:3  U001:6
   2 #002       U002:6  R003:1
   3 #003       R003:2  D005:1
   4 #004       U002:11 U001:8
   5 #005       U002:8  R004:1
   6 #006       R004:2  D006:1
   7 +5V        CONN:9  U002:2
                D005:2  U002:12
                D006:2  U001:14
                U002:14
   8 GND        CONN:11 U001:7
                U002:7
   9 I1         CONN:1  U001:1
  10 I2         CONN:2  U001:2
  11 I3         CONN:3  U001:4
  12 I4         CONN:4  U001:5
  13 I5         CONN:5  U001:9
  14 I6         CONN:6  U001:10
  15 I7         CONN:7  U001:12
  16 I8         CONN:8  U001:13
  17 NC         U002:5  U002:9
  18 RESET      CONN:10 U002:1
                U002:13
```

Figure 8-2. Part and Net List

We will begin the layout by loading the initial board file created with the EDIF processor. Select the Load entry on the File menu. Since we already have a board loaded, we will be questioned as to whether or not we wish to lose the changes made to the current board. Answer Yes when prompted.

Enter "MYPCB" and *ENTER* to load the initial PC board for our tutorial.

8.7 Optimizing Board Placement

The placement of the parts on the board, as it comes from the EDIF processor, is not optimized. Our first step in completing the layout is to shift the components around on the board to make the layout of the net lines easier and more electrically efficient. Also, any components which must be restricted to a given location are placed at this time.

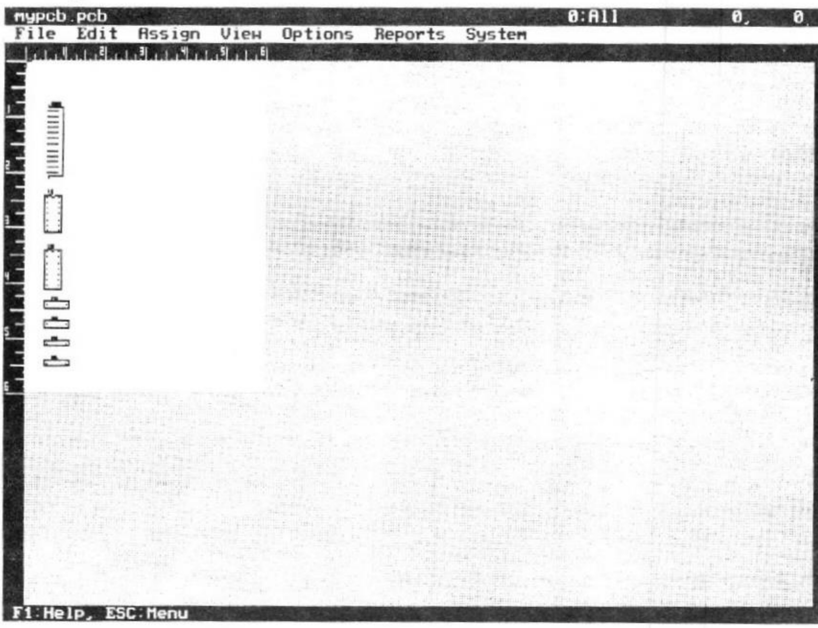


Figure 8-3. Initial EDIF Board

Select the Components command from the Edit menu. Move to location (700,900) and *ENTER* to select component "EC-12". Move to location (150,1000) and *ENTER* to place the connector. *ESC* to release the component. This is how components are moved about on the board.

Now, move to (550,2450) and select component "U001". Move to (650,600) and *ENTER* to place it. Now press the *N* key to display a rat's nest of connections from the selected component. This is used to aid in placing components. By being able to see how a component connects to other components, it is easier to optimize the component placement. Press the *N* key again to turn off the rat's nest display. *ESC* to release the component.

Use the method just learned to place the remaining components as described below. If you like, use the *N* command to show the rat's nest as the components are placed. In actual practice, you can select a component by placing the cursor anywhere within its area. For agreement with the rest of the tutorial, use the following exact coordinates as you make the placement. The coordinates given are pin one of that component.

In the following exercises, you may want to use the "locate cursor" command. This command may be used to directly position the cursor to an exact coordinate. To use this command, just press *F4* and enter the x and y coordinates.

```
U2   at (550,3450) to (650,2100).
R003 at (550,4750) to (1500,1150).
D005 at (550,4400) to (2500,1150).
R004 at (550,5450) to (1500,1600).
D006 at (550,5100) to (2500,1600).
```

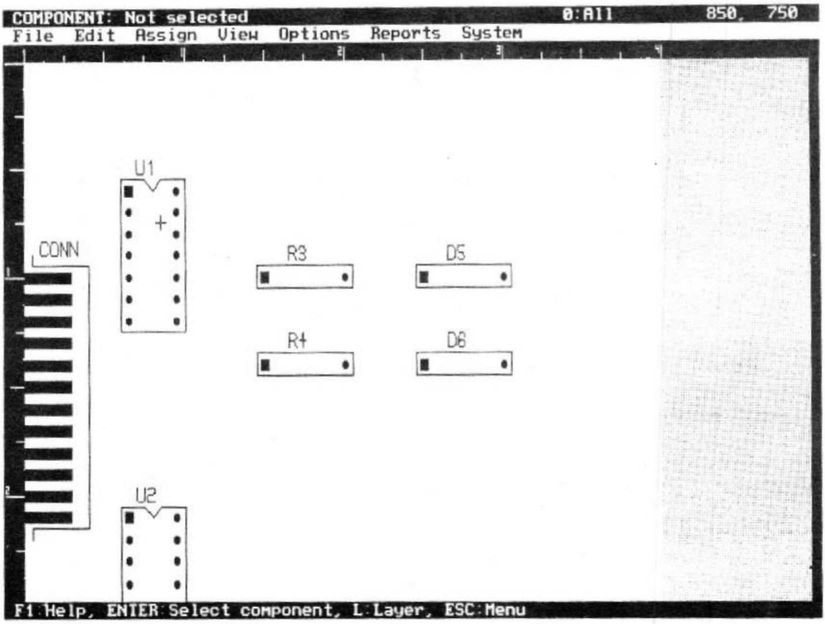


Figure 8-4. Board after Placement

8.8 Set Board Size

Now that the board is placed, we will set the board to the specified size. Select the Edit/Board Size command. Select Xhigh and *ENTER*. Set the value to 4000 (1 inch is 1000 mils). Likewise, set Yhigh to 4000.

8.9 Add Power/Ground Layers

While we are in the board size function, we will also add the power and ground layers. Select the # layers and change the value from 4 to 6.

Now, select the Layer types function. Select layer 5 which has type "none" and change its type to Power/ground. Do the same for layer 6. Press the *ESC* key twice to complete the board size function.

At this point, we have added two layers to the board and designated them as power/ground types. Now, we will assign a net name to each of these two layers. Select the Gnd/pwr layer command on the Assign menu. Select layer 5 and enter "+5V" for the name. Likewise, assign the net name "GND" to layer 6.

8.10 Routing the Board

We are now ready to add the routes that connect the pads together to make the desired circuit. Once again we will work directly from the net list given in Figure 8-2. Select the Lines command from the Edit menu.

First, we will choose the desired width of the route lines. From our specification, this is 12 mils. Enter *O* and *W* (for option/width), and set the value to 12. For this board, we will use the component-side for the mostly horizontal routes, and the solder-side for the mostly vertical routes.

From our list, we will route the #001 net first. This net connects from U002 pin 3 to U001 pin 6. Move the cursor to U002 pin 3 (650,2300). Set the layer to the solder-side. Press the *N* key. This will display a prompt-line to the nearest pad that this pad connects to. This is for reference only, the line is not part of the board; however, it is extremely helpful in routing boards by indicating what pads need to be connected together. The line may be turned off by pressing the *N* key again. Press the *N* key several times to flash the path. Leave the line turned on for the next step.

We are now positioned on one end of the route to be made, and we have an indicator showing where we are going. *ENTER* to begin the route. Now, move to (750,2300). Notice that the line we are about to draw is displayed. *ENTER* to add this segment to the board. Now, move to (750,1100) and *ENTER*. Move to (650,1100) and *ENTER*. This completes this route and this net. *ESC* to terminate this line. Press *N* to remove the net-prompt line.

Now, we will route the next net, #002 which connects U002 pin 6 to R003 pin 1. Move the cursor to the location of U002 pin 6 (650,2600) and press *N*. Route this net as follows: Use the *T* key to toggle to the component-side. *ENTER* to start the net. Move to (800,2600) and *ENTER*. At this point we will add a via and switch to the other side of the board by pressing *ENTER* again. Move to (800,1150) and *ENTER*. *ENTER* again to add a via here. Move to (1500,1150) and *ENTER* to complete this route. Enter *ESC* to terminate the route and *N* to remove the net-prompt. Net #002 is now complete. Now, let's guide you through one more net, #003. Move to U002 pin 6 at (2000,1150). *N* to display the path prompt. This is an easy one! *ENTER* to start the line, move to (2500,1150), *ENTER* to add the line, and *ESC* to complete this net.

This is how nets are routed. See how the net names assigned to pins really pay off!

Now, use the net list and proceed to complete the rest of the nets on this board, just like we routed the first three nets. Warning! When you make a route to one of the connector pads, remember that these pads are only on the component-side of the board. Make sure your connection to the pad is also on the component-side. Also, the connector pads for nets +5V and GND may be connected to the power layers by routing a short line and ending it with a via. The via will connect to the power plane of that net name.

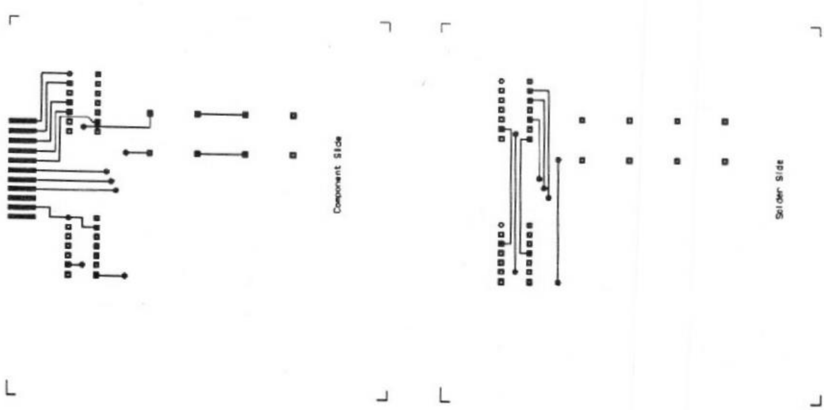


Figure 8-5. Component & Solder Sides

8.11 Adding Text

Additional text, other than the component instance numbers, is often added to identify the layer of the mask, the company name, copyright notice, etc.

Select the Text command form the Edit menu. Set the working layer to the component side layer. Enter *OS* (option-size) and set the text size to 62 mils (1/16"). Move to (1000,300) and *ENTER* to add a text string. Enter "Copyright (c) 1988 by Me". The text will appear on the board at the location of the cursor and have attributes which match the current settings displayed at the bottom of the screen.

Enter *OA* (option-angle) and select Up for the angle. Move to location (3500,2000). *ENTER* and add the string "Component Side". Set the layer to Solder-side. *ENTER* and add the string "Solder Side". Likewise, add the names for the Power, and Ground layers.

You may add any additional text as desired. If you add text on a conducting layer, be careful in placing it so as to not make a short. Set the angle, size, and mirror factors as desired from the option menu. To change the factors for an existing text string, set the parameters to the new desired values, place the cursor on the string and enter *U*. Change the string if required and *ENTER* to complete the update of the text.

To delete a text string, place the cursor on the text, set the layer to the layer for the text and *DEL*.

8.12 Crop Marks

To complete the board, add alignment marks in each of the board corners. Use a line width of 1 mil and apply the lines to the all layer.

8.13 Reports

Pull-down Reports from the main menu. Generate a parts list a net list (pads only option), and a Summary report. If you have a printer, enter "PRN" for the device name. This will send the reports directly to the printer. Compare these reports with the input net and parts lists. They should agree.

In actual practice you would not have to make this comparison manually. The PC-BRIDGE utility (not included in the evaluation kit) would be used to verify that the PC board layout agrees with the schematic design data. The utility is also be used to update the design information (component and pin/net names) for a board when changes to a schematic are made.

8.14 Saving the Board

Select save As from the File menu. Enter "MYBOARD" and *ENTER*. ***The evaluation version of the program will not actually save the board to a disk file.***

8.15 Exit the Editor

We are finished with the board for now. Select Quit on the File menu to exit the program.

9 PC Board Auto-router

In actual practice, the EZ-ROUTE auto-router is used to automatically route the connections for a board. Although the router cannot route every board to 100% completion, it does perform the vast majority of the board layout.

Before a board is routed, the board is first "scheduled". The schedule program gets a board ready for routing by determining which points on the board need to be connected. The schedule may be adjusted, or entered directly, with the PCPRO-II editor.

Once the board is prepared, the auto-router uses artificial intelligence to route the connections of the board. The auto-router will preserve any nets that you have pre-routed. Also, the auto-router may be interrupted while it is running. At this point the PCPRO-II editor may be used to manually make adjustments to the board. The board may then be resubmitted to the auto-router to continue the remainder of the routing.

For a demonstration of the auto-router, run the AMSCAD supervisor program and select the Auto-router/Demo command.

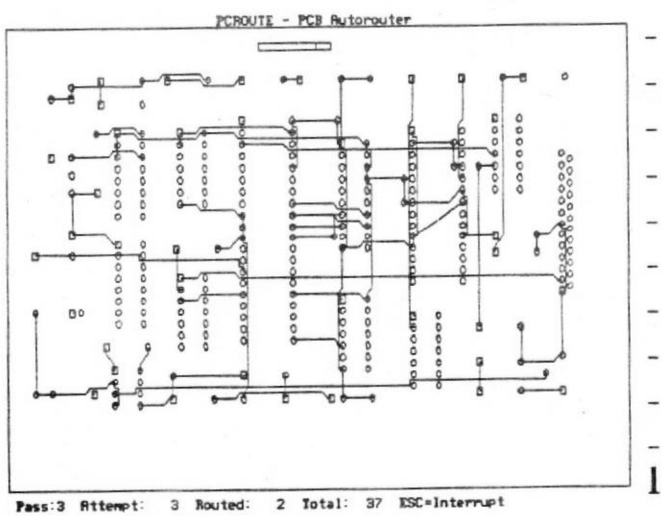


Figure 9-1, Auto-router in Action

10 Summary / Ordering Information

Thank you for taking the time to evaluate this software package. We hope it has been an enjoyable and educational experience. If you find that you still have questions, give us a call at (305)-975-9515.

To order the EZ-ROUTE system, or any of its subsystems, just call toll free 1-800-9PCFREE to enter your order. Don't delay; order today!

EZ-ROUTE	Integrated CAE system includes AMSCAD supervisor, CELL-II schematic system, EDIF schematic processor, PCPRO-II board layout system, PCROUTE auto-router, PLOT and PCPLOT hard copy utilities, EDFONT font editor, PCBRIDGE convert/update program, part and symbol libraries. Does not include PCCHECK.
CELL-II	Includes CELL schematic editor, EDIF schematic processor, PLOT hard copy utility, and EDFONT editor, part and symbol libraries.
PCPRO-II	Includes PCPRO layout editor, PCPLOT hard copy utility, EDFONT editor, PCBRIDGE convert/update program, part footprint library.
PCROUTE	Includes PC board auto-router and PCSCHED scheduler.
PCCHECK	Includes PC layout design rule checker.

A - APPENDIX: Additional Print/Plot Information

Hard copy output for schematics is generated with the CELL program. PCB masks are plotted with the PCPLOT program. These programs may be run directly or from the AMSCAD supervisor.

These are high performance programs, capable of generating output faster than an output device can plot the information being sent to it. For plotters which are connected to your system via a serial port, you must use a cable which has the proper handshaking wires in it. This allows the plotter to signal the computer to wait until it catches up. Standard RS232C cables have the proper wires; however, homemade three-wire cables do not.

If you do not have a proper cable, the plotting will initially be correct. Once the buffer of the plotter becomes full, the pen will appear to make random movements and draw random lines on the plot. If it is a small plot, you may sneak by.

The default serial port settings for the EZ-ROUTE programs are 9600 baud, no parity, and one stop bit.

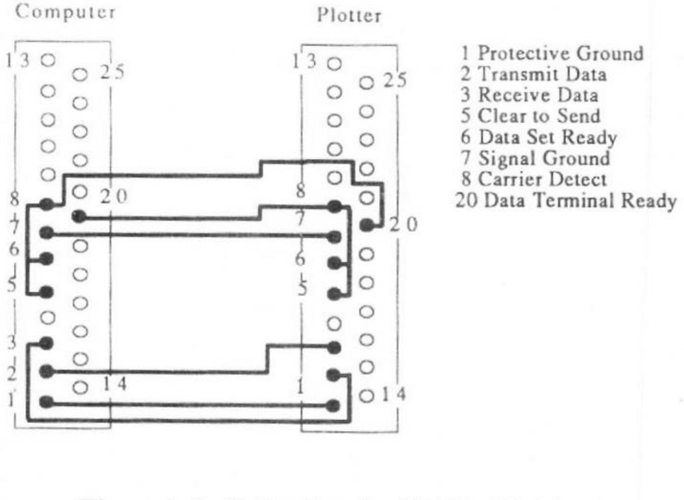


Figure A-1. Cable Circuit with Handshaking